

User Manual

MEGA S

Dear customer,

Thank you for choosing **ANYCUBIC** products.

Maybe you are familiar with 3D printing technology or have purchased **ANYCUBIC** printers before, we still highly recommend that you read this manual carefully. The installation techniques and precautions in this manual can help you avoid any unnecessary damage or frustration.

More information please refer to :

1. <http://www.anycubic.com/>

ANYCUBIC website provides software, videos, models, after-sale service, etc.

Please go to our website to report any issues and we are likely to answer or solve all the questions for you!

2. Facebook page and Youtube channel as shown below.



ANYCUBIC website



Facebook page



Youtube channel

Team **ANYCUBIC**

Safety instruction

Always follow the safety instructions during assembly and usage, to avoid any unnecessary damage to the machine or individual injury

 Please contact our customer service first if you have any issue after receiving the products.

 Be cautious when using the scraper. Never direct the scraper towards your hand.

 In case of emergency, please immediately cut off the power of **ANYCUBIC** 3D printer and contact the technical support.

 **ANYCUBIC** 3D printer includes moving parts that can cause injury.

 It is recommended to wear protection glasses when cleaning/ sanding the printed models to avoid small particles contacting eyes.

 Keep the **ANYCUBIC** 3D printer and its accessories out of the reach of children.

 Vapors or fumes may be irritating at operating temperature.
 Always use the **ANYCUBIC** 3D printer in an open and well ventilated area.

 **ANYCUBIC** 3D printer must not be exposed to water or rain.

 **ANYCUBIC** 3D printer is designed to be used within ambient temperature ranging 8°C-40°C, and humidity ranging 20%-50%. Working outside those limits may result in low quality printing.

 Do not disassemble **ANYCUBIC** 3D printer, please contact technical support if you have any question.



Contents

Technical Specification -----	1
Packing list -----	2
Product Overview -----	3
Menu Directory -----	4
Installation -----	8
Install frame-----	9
Wiring -----	10
Leveling -----	12
Printing test -----	17
Filament in-----	17
Test print -----	18
Driver installation -----	20
Cura installation -----	21
Introduction to slicing software -----	22
Manipulate 3D model in Cura-----	23
Cura settings-----	25
Print online-----	30
Print offline-----	31
Printing -----	31
Manual filament change -----	33
Resume from outage -----	34
FAQ -----	36

Printing

Technology:	FDM (Fused Deposition Modeling)
Build Size:	210×210×205 (mm ³)
Layer Resolution:	0.05-0.3 mm
Positioning Accuracy:	X/Y 0.0125mm, Z 0.002mm
Extruder Quantity:	Single
Nozzle/Filament Diameter:	0.4 mm/1.75mm
Print Speed:	20~100mm/s (suggested 60mm/s)
Travel Speed:	100mm/s
Supported Materials:	PLA, ABS, HIPS, Wood

Temperature

Ambient Operating Temperature:	8 °C - 40 °C
Operational Extruder Temperature:	max 260 °C
Operational Print Bed Temperature:	max 100 °C

Software

Slicer Software:	Cura
Software Input Formats:	.STL, .OBJ, .DAE, .AMF
Software Output Formats:	GCode
Connectivity:	SD card; USB port(expert users only)

Electrical

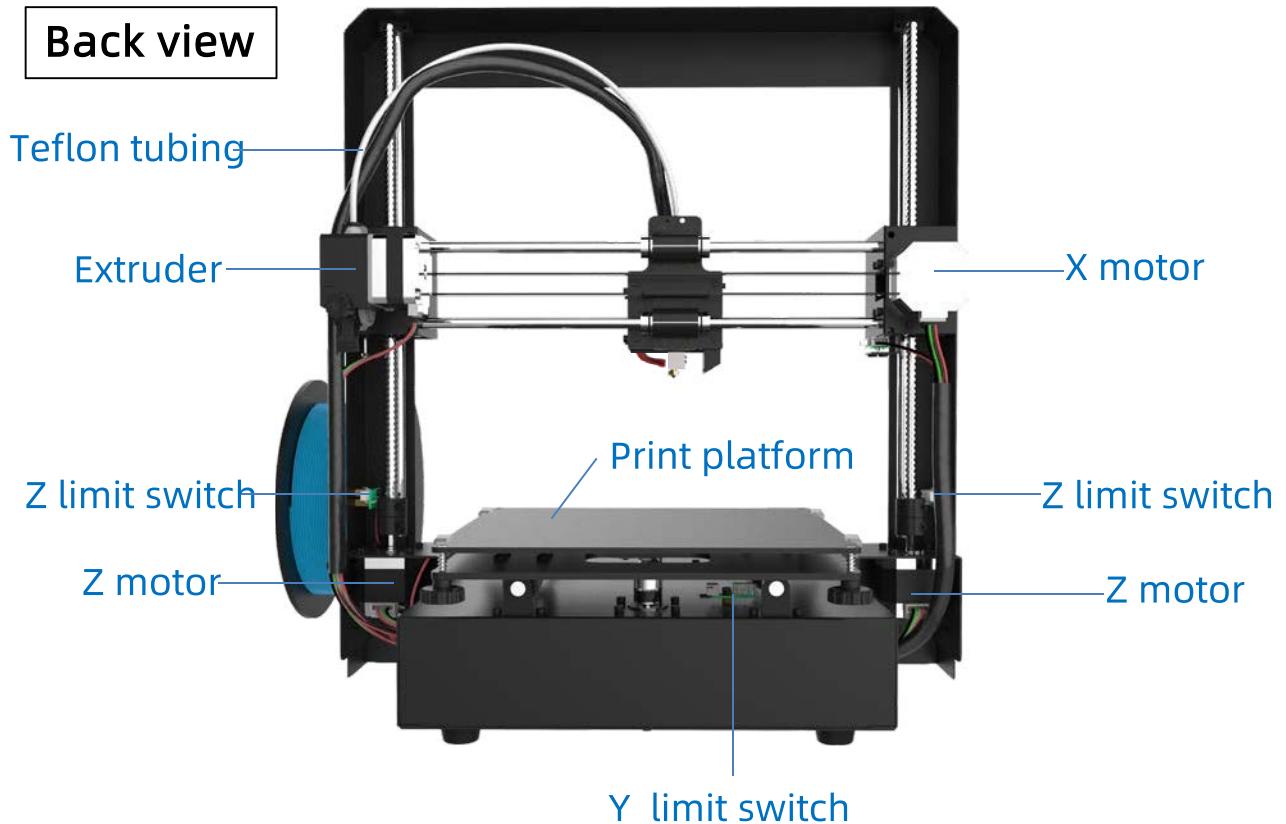
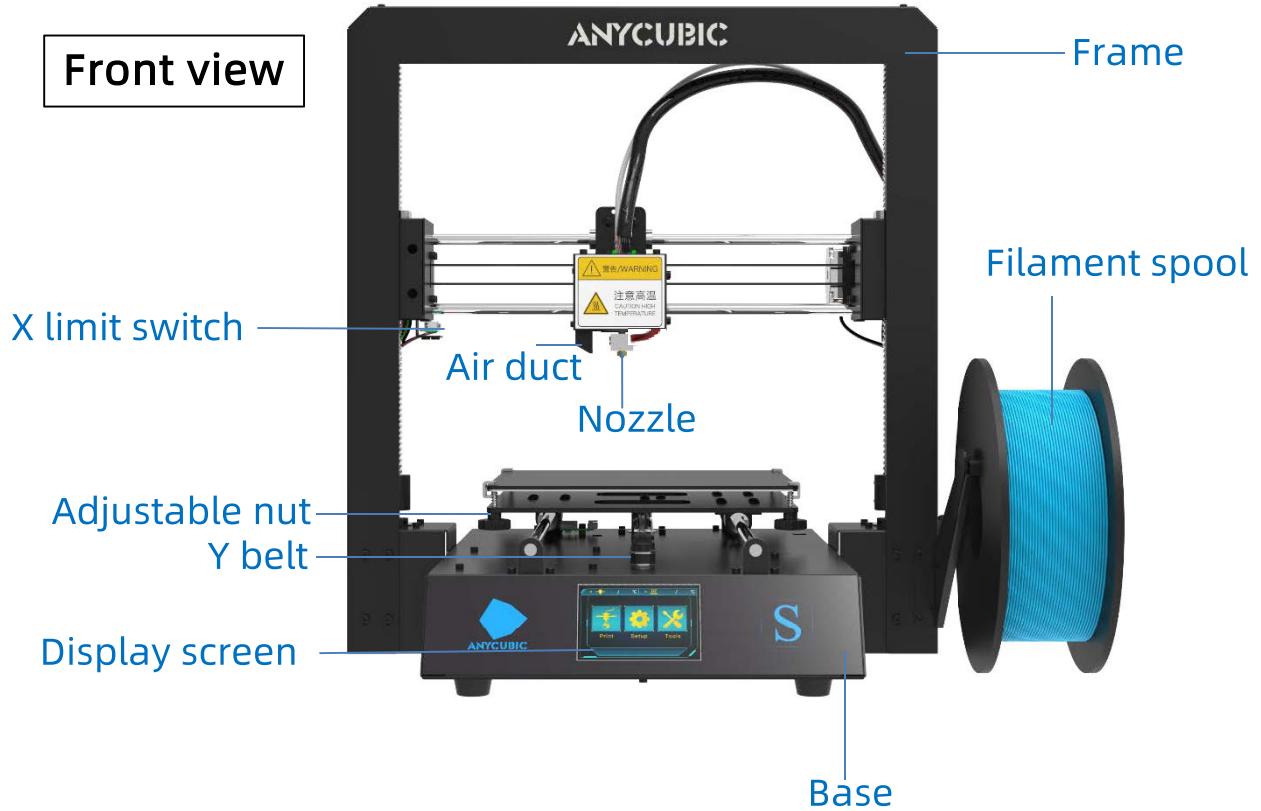
Input rating:	110V/220V AC, 50/60Hz
Working Voltage:	12V DC

Physical Dimensions

Printer Dimensions:	405mm×410mm×453mm
Net Weight:	~11kg

Packing list

		
	M5*8 screw 10PCS	Plier 1PCS
		
ANYCUBIC MEGA S	Filament holder 1PCS M3*5 screw 2PCS	Filament 1PCS
		
Assembly Instruction 1PCS	After sale service card 1PCS	Extra limit switch 1PCS
		
Power cord 1PCS	Data cable 1PCS	Shovel 1PCS
		
Memory card 1PCS Card reader 1PCS	Extra print head 1PCS	Tool kit 1PCS



Menu Directory

Home menu



Print

Setup

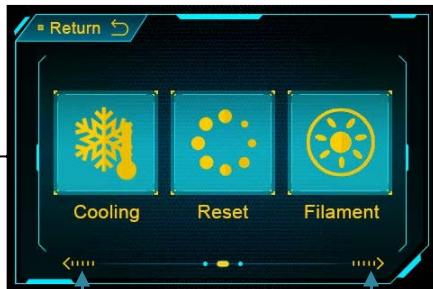
Tools



Setup



Tools



Tools

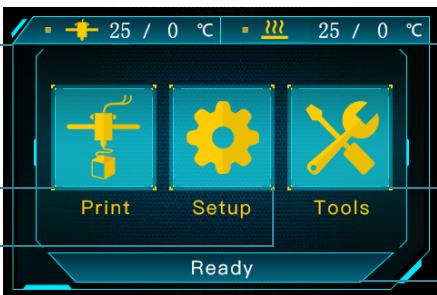


Previous page

Next page

Home menu

Nozzle Temp /Target Temp



Heated bed Temp/Target Temp

Enter the print list

Enter the setup list

Enter the tools list

Printer status

Print

Return to the home menu



Page up

Page down

Refresh the list

File list

Print the selected files
in memory cardResume form outage (only valid
for offline print via memory card)

Setup

Language: Change language (English/Chinese)**Temperature:**

Reduce nozzle Temp

Raise nozzle Temp

Click to set nozzle
Temp(170-260°C)Click to set heated bed
Temp(0-100°C)

Current nozzle Temp

Current heated bed Temp

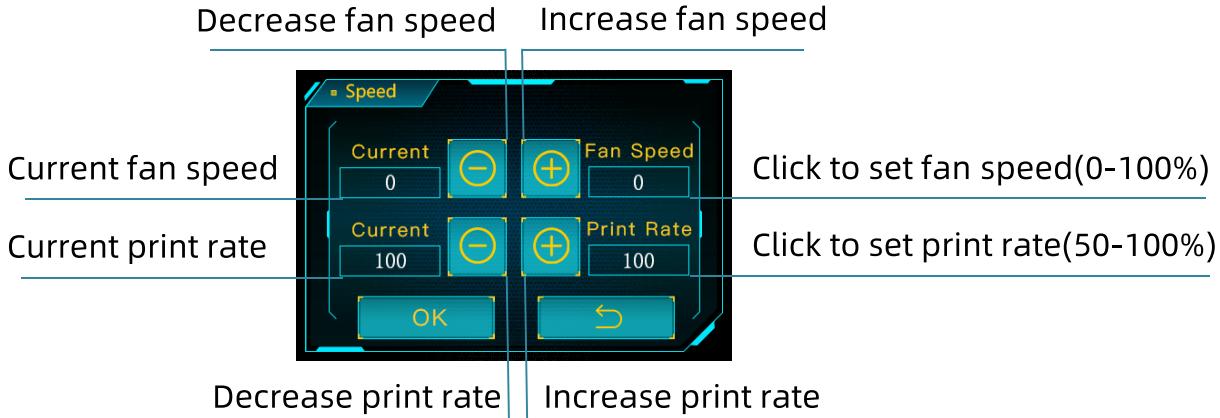
Reduce heated bed Temp

Raise heated bed Temp

Menu Directory

Motor: Disable all motors (only valid when machine is not printing)

Speed:



Status: (the following with * is valid only for offline printing , i.e. print from memory card)



Voice: Turn on/off the screen sound

Tools

Home: (only valid when machine is not printing)

Return to previous menu

Click to home X

Click to home Z



Click to home Y

Click to home All

Menu memory

Axis: (only valid when machine is not printing)



- Move left/right X axis by 0.1/1.0/10mm
- Move backward/forward Y axis by 0.1/1.0/10mm
- Move down/up Z axis by 0.1/1.0/10mm

Return

Speed mode for axis move Low/Medium/High

Preheat: (only valid when machine is not printing)

Nozzle Temp/Target Temp

Click to preheat PLA



Heated bed Temp/Target Temp

Click to preheat ABS

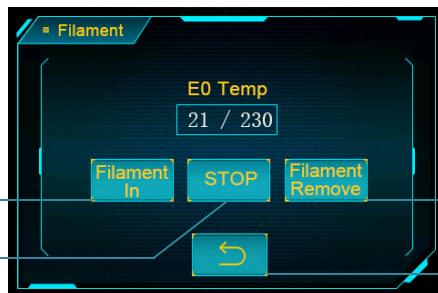
Return

Cooling: Cut off the power to hot-end and heated bed (only valid when machine is not printing)

Reset: Popup window to decide if reboot the mainboard

Filament: (only valid for offline print)

Automatically heating to the setting temperature for filament in
Stop to filament in/remove



Automatically heating to the setting temperature for filament remove

Return

Help: Basic description of the Menu

About: Information about the product

Installation section

1. Installation section contains: ①Install the frame ②Wiring ③Install the filament holder and filament
2. Be cautious during assembly as some parts may have sharp edges.
3. It is suggested to use a flat desktop and place the parts in an orderly manner for quick assembly.
4. The color of some parts may be different from what in the manual, but the assembly is the same.
5. Firmware has been pre-uploaded to the motherboard. After completing the assembly, please load the filament and level the platform then you could start the first test print.

1. Install frame

(1) Fig.1 , carefully lift the base to fit into the frame and fix them by 8 pieces of M5*8mm hex cap screws as shown in the red boxes.Fasten the screws when all the screws are pre-installed.

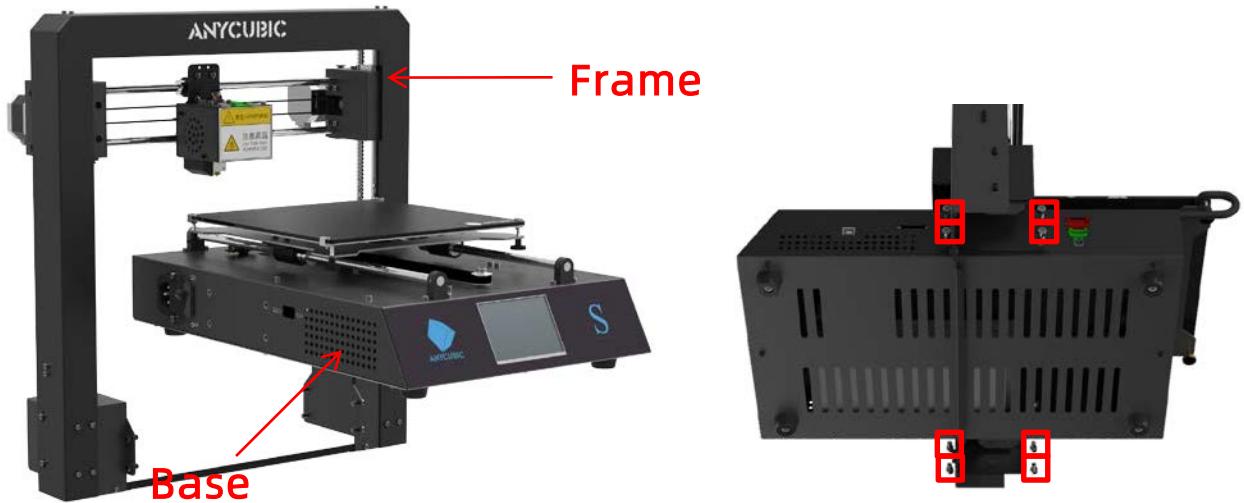


Figure 1

(2) Use two M3*5 screws to install the filament holder, then screw out two M3*5 screws that holding the frame to the base and fix the filament holder to the frame by these two M3*5 screws as shown in Fig.2.

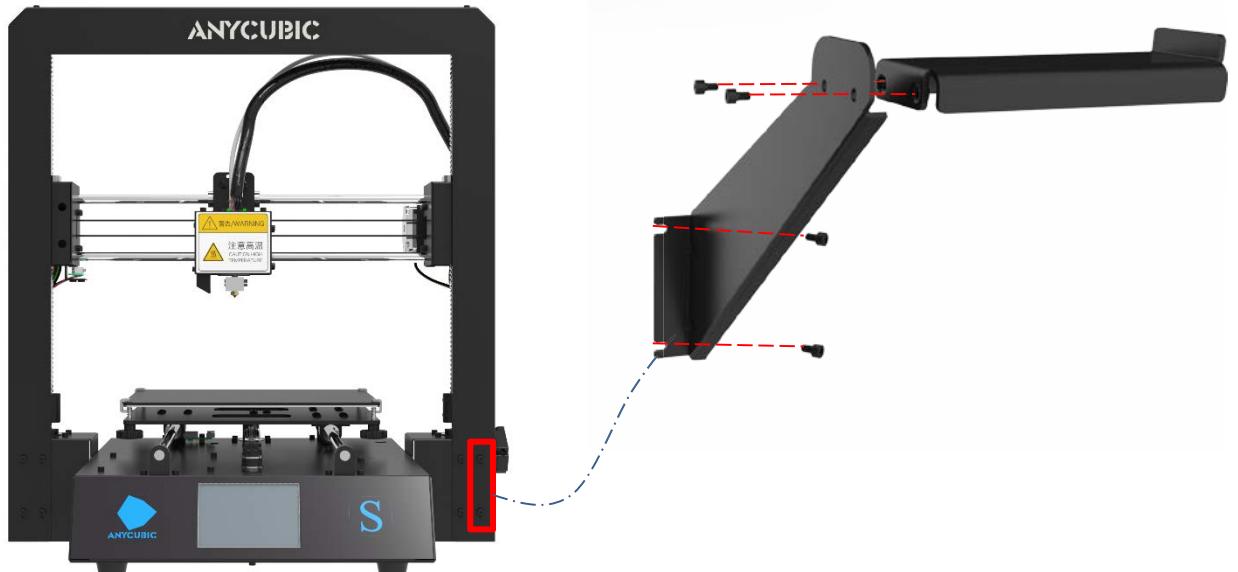


Figure 2

Installation section

2. Wiring

(1) Fig.3, select the correct voltage mode according to your local voltage ratings (110V/220V). The switch is inside the bottom left of the base and 220V is default. Allan keys can be used to move the switch inside.

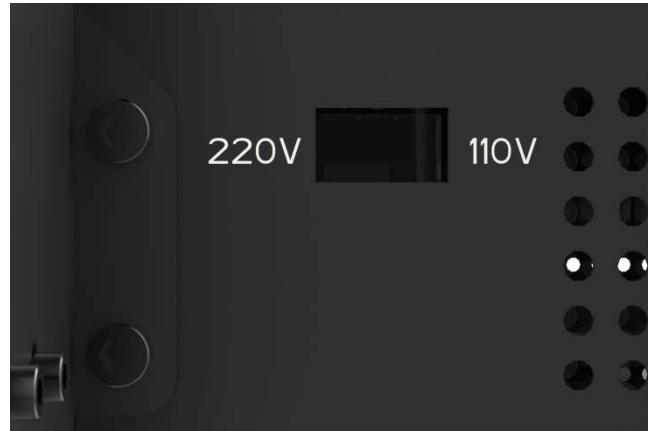


Figure 3

(2) There are 3 ports with different color (Red/Green/Black) at the bottom right side of the base, and there are 3 different color cable connectors respectively. Accordingly, insert those connectors to the ports by the same color, as shown in Fig.4.

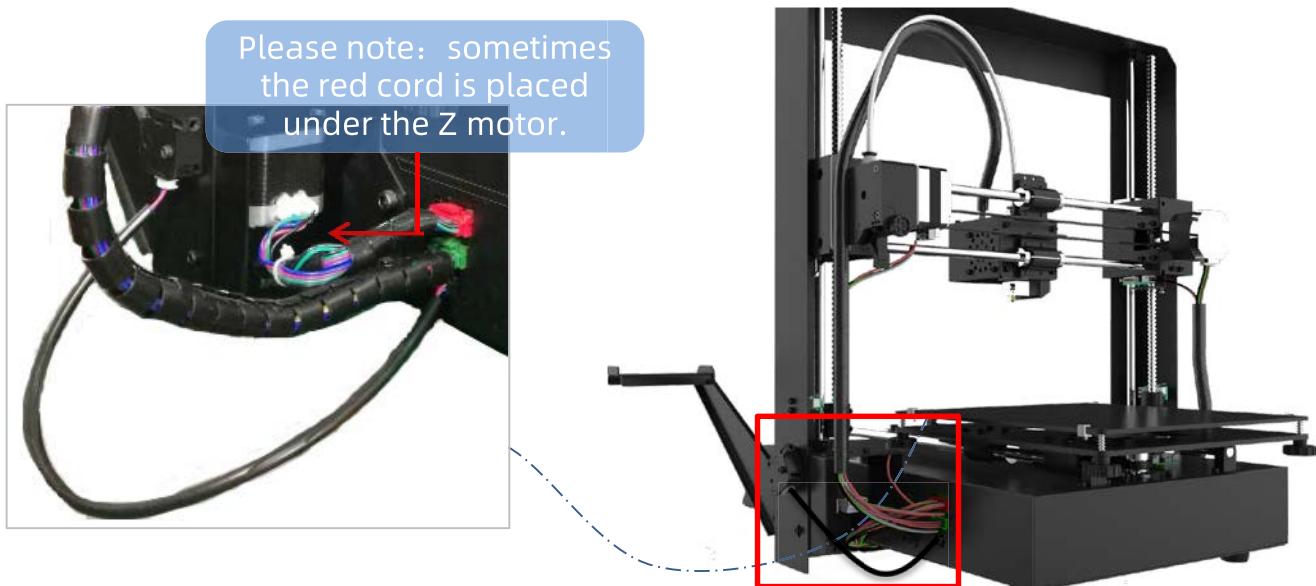


Figure 4

- > Make sure the connectors are well inserted, and no pins are bent inside.
- > Wrong or loose connection would lead to malfunction of the machine.

Installation section

(3) Fig.5, customers may notice there is a ring of zip tie attached just below the plastic ring of the quick connector. Do not cut it off. Only cut this zip tie when swapping or repairing the hotend.



Figure 5

Please note: every units of the printer have been inspected and tested for actual printing. Therefore, in some cases, there might be very small marks left on the print head or on the heated bed. Those will not affect the printing quality and those means the printer has been tested for the quality. Meanwhile, we provide an extra hot end in case you need to replace it in the future. Thank you very much for your kind understanding.

Leveling

Leveling the platform is a key step in 3D printing. Please follow the leveling procedures below to achieve proper leveling so the printed models could stay firmly on the printing platform and deliver good results. Otherwise, if the distance between the nozzle and printing platform is too large, the printed product will not stick properly to the platform, and if the nozzle is too close to the printing platform, the filaments would not been extruded properly from the nozzle and causing clog or even damage to nozzle or platform.

Step 1. Double check all wirings are OK, and then connect the machine to the power outlet by power cord. Switch on the machine.

Make sure: (1) the nozzle is clean without filament residue, (2) the printing platform is clean, otherwise it will affect the leveling accuracy.

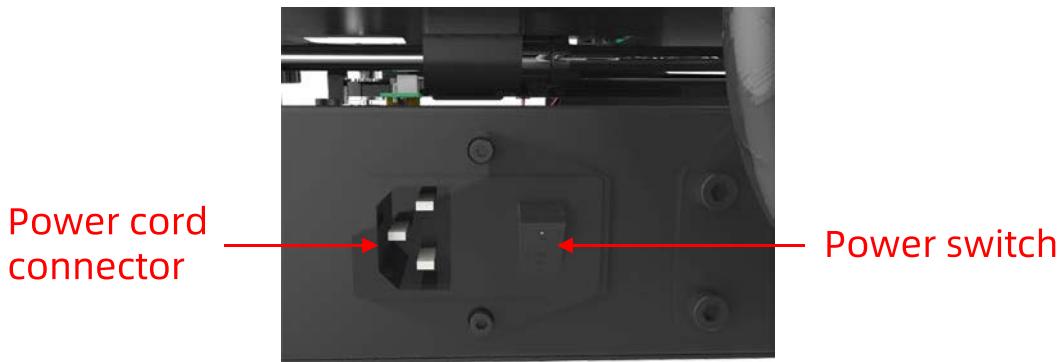


Figure 6

Step 2. As shown in Fig.7, on Home Menu, click “Tools”-->“Home”-->“Home Z”. **Lock the Z axis to prevent the Z axis from falling during leveling.**



Figure 7

Step 3. Place a piece of A4 paper on the lower left corner of the printing platform, and manually move the printing head and platform back and forth to let the printing head to be above the lower left corner of the paper, as shown in Fig.(8).



Figure 8

Step 4. Tighten or loosen the adjustable nut underneath to adjust the distance between the nozzle and the platform to about the thickness of the paper(~0.1-0.2mm, the nozzle just touches the platform), as shown in Fig.9. You need to adjust the adjustable nut until you feel the drag resistance when pulling the paper back and forth. (Note: "feel the drag resistance" means the paper can be moved, but with resistance)

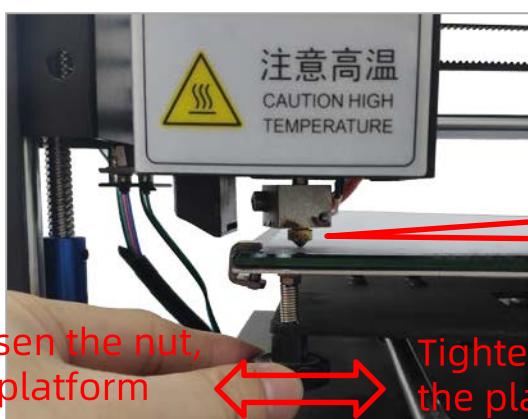


Figure 9

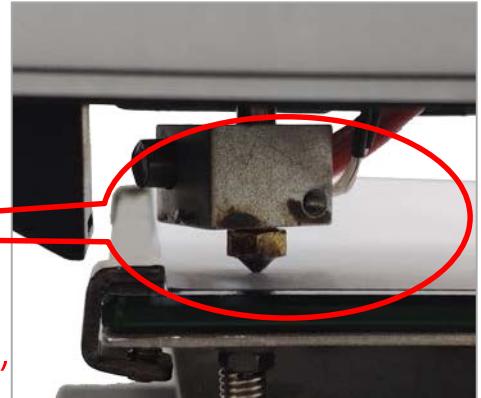


Figure 10

Note: Do not press on the platform when adjusting the nut, otherwise it will be affecting the leveling accuracy.

Leveling

Step 5. Follow step 4 to level the other three corners and the center of the platform, as shown in Fig.11.

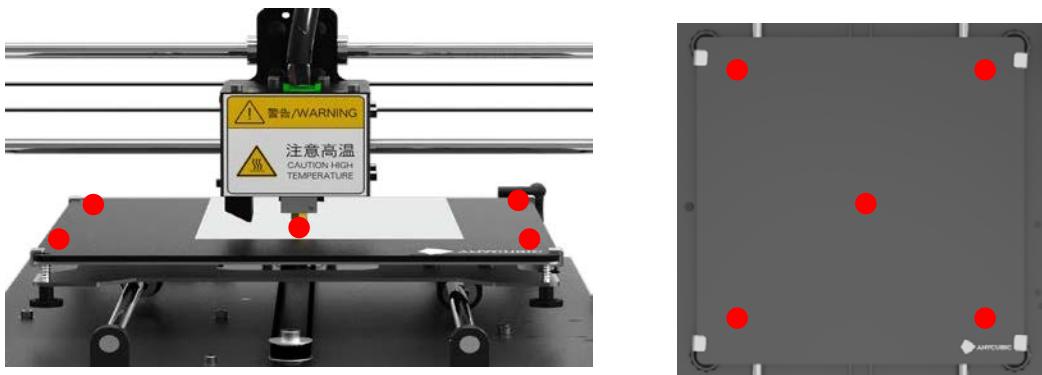


Figure 11

Step 6. Move the printing head and platform at the same time, so that the printing head can be moved in diagonal order, as shown in Fig.12. In the process of moving, check whether the distance between nozzle and printing platform is about a piece of paper thin or not. You may need to adjust the 5 points of the platform 1 or 2 times to check the leveling result.

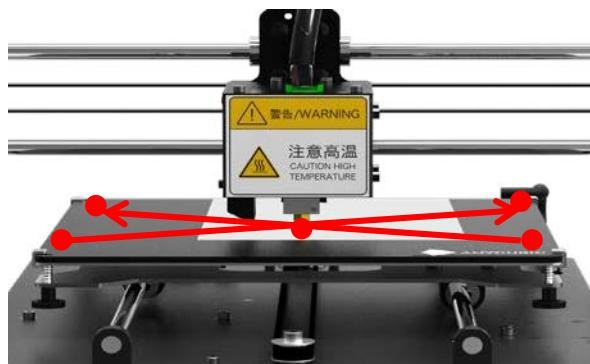


Figure 12

Note: Do not let the nozzle rub against the printing platform directly during the whole leveling process. When moving the nozzle, paper must be placed on the platform to prevent the platform from being scratched by the nozzle.

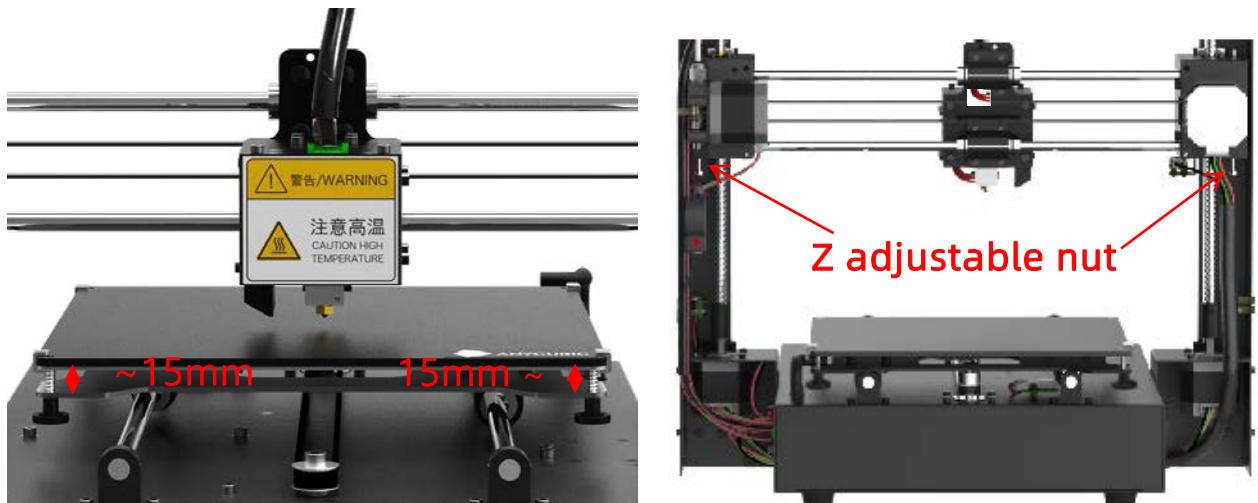
Tips: The printing platform has the characteristics of high temperature resistance, long service life, good adhesion and easy removal of prints. Its flatness is in the range of 0~0.2mm variation, means every piece of the platforms has been tested by a 0.2mm feeler gauge on a flat marble table.

Supplements to leveling:

In some rare cases, after “Home All”, the nozzle can be still much lower than the platform, even after fully tighten the 4 nuts underneath. On the opposite, sometimes the nozzle is still too high from the platform, even after fully loosen the 4 nuts underneath.

How to solve this:

- (1) Raise the nozzle by click “Tools”--> “Axis” --> “10” on +Z column, adjust the 4 nuts under the platform, let the height in-between the support plate and the heated bed is ~15mm for all the 4 corners.
- (2) As shown in **Fig. 13**, at both ends of X axis, there is a Z adjustable nut. The lower tip of Z adjustable nut can trigger the Z end stop when Home (going down), and ‘tell’ the machine Z axis is getting to zero and stop moving.

**Figure 13**

- (3) Therefore, tighten Z adjustable nut by X mm if nozzle is lower than the platform (X is defined by how much the nozzle is under the platform), while loosen it by Y mm if nozzle is too high from the platform (Y is defined by how much the nozzle is above the platform). It may need adjustment for few times.

Leveling



(4) Click “Tools”-->“Home” -->“Home all” to verify the results. After this, please level the platform again from **Step 3**.

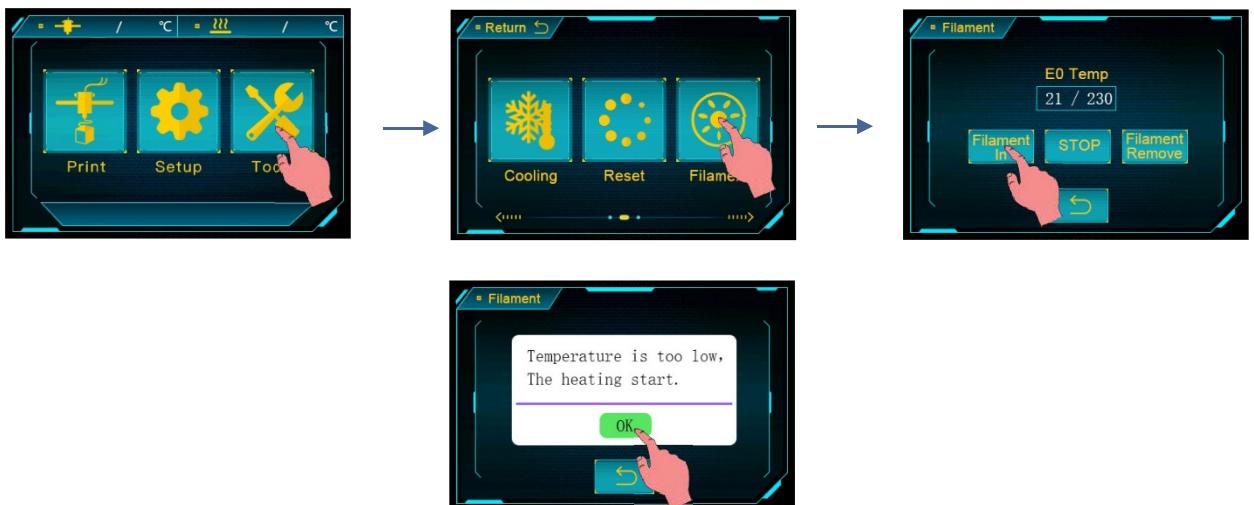
Printing test

1. Filament in

(1) Return to the home menu, click "Tools"→"Axis"→"10+Z" 10 times to rise the print head.

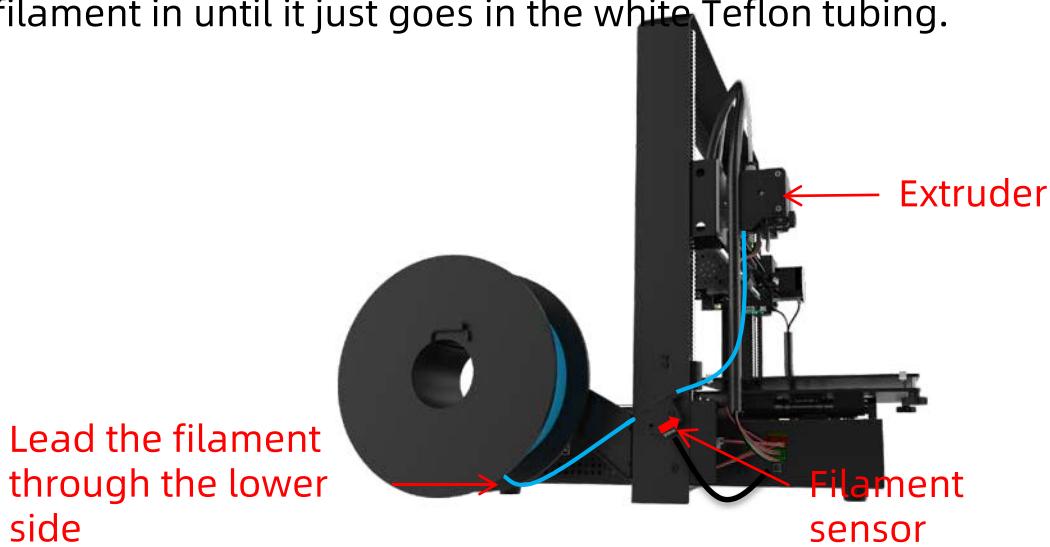


(2) Return to the home menu, click "Tools"→"Filament"→"Filament in", and the interface as shown below will pop up, click "OK".



(3) Place the filament on filament holder, please note the direction of filament.

Straighten the end of filament, pass the filament through filament sensor, and then press the handle on the extruder and push the filament in until it just goes in the white Teflon tubing.



Printing test

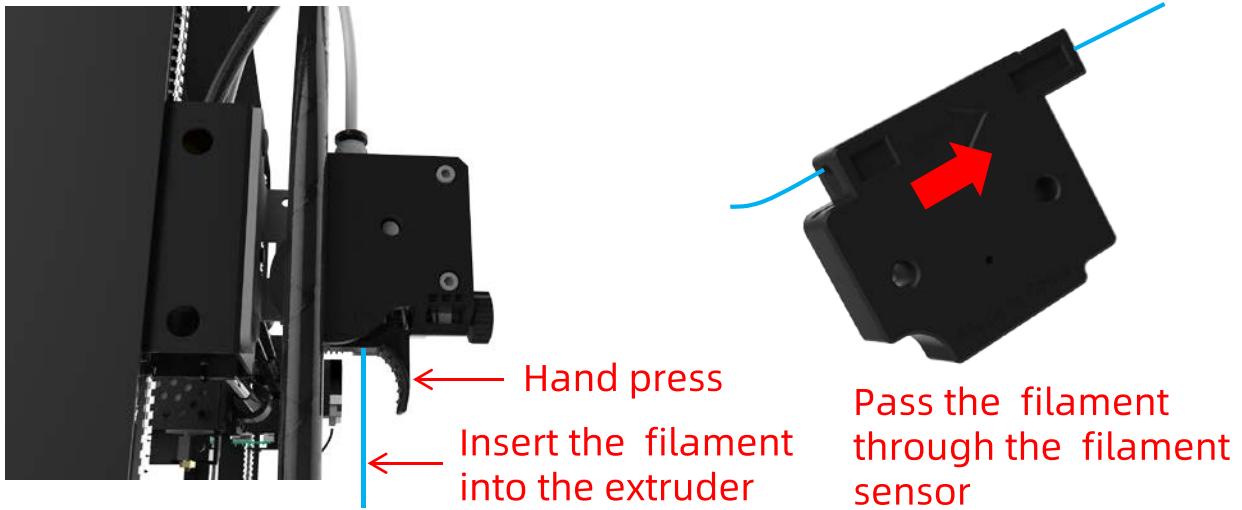


Figure 14

(4) As shown in Fig. 15, slide the print head to the left, when the target temperature reached, click “Filament in” again, the extruder will automatically feed the filament in till the filament is melted through the nozzle. Now, click “Stop” on the screen, you may use tweezers to clean the nozzle.



Figure 15

2. Test print: insert the SD card (back side facing up) into the SD card slot on the printer base. Click on the Home Menu “Print” to enter the file list (Fig.17) . There is a printable test file included -- “owl_pair” (author: etotheipi, www.thingiverse.com), and please print it to verify the leveling results.



Figure 16



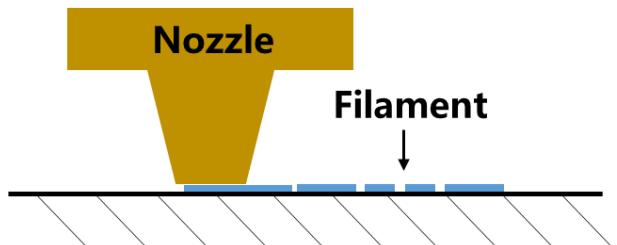
Printing test

There might be 3 kinds of results for the first layer of the test prints.

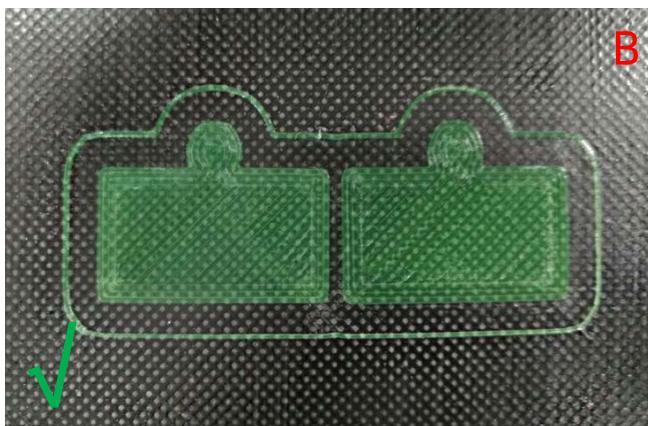
A: Nozzle too close, lack of extrusion, the nozzle rub against the platform. Slowly tighten the corresponding nuts underneath by half circle or level again.



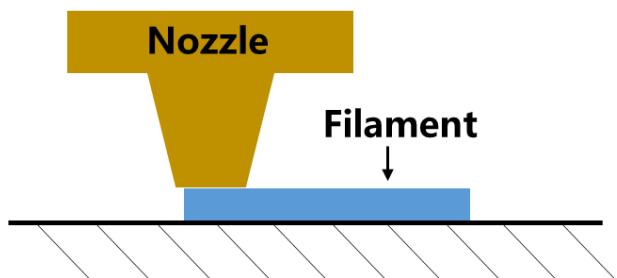
Nozzle too close



B: Proper nozzle height, good extrusion and adhesion.



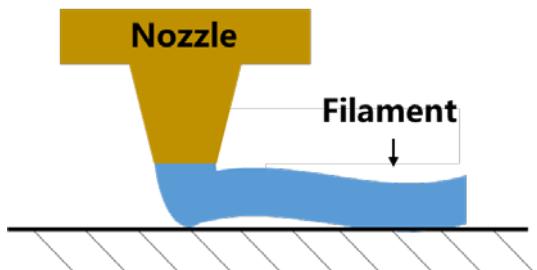
Proper nozzle height



C: Nozzle too high, Large gap, filaments are not even adhere to the platform. Slowly loosen the corresponding nuts underneath the platform by half circle or level again.

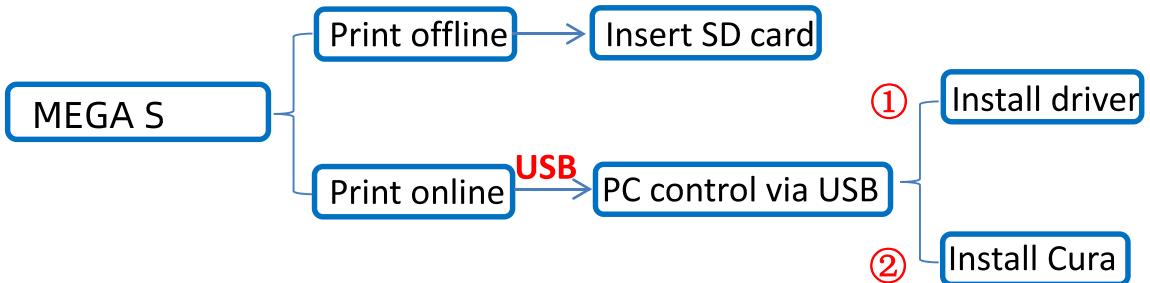


Nozzle too high



Driver installation

There are two operation mode for MEGA S 3D printer, print offline (via SD card) and print online (controlled by PC via USB). **Print offline:** After leveling the platform, insert SD card, click “Print” at the Home menu and select a file to print. **Print online:** Install driver to bridging PC and machine, and install Cura to control the machine via USB port.



Generally, it is suggested to use Print offline to minimize the noisy signal via USB. The preparation steps for Print Online are shown as below.

Driver installation

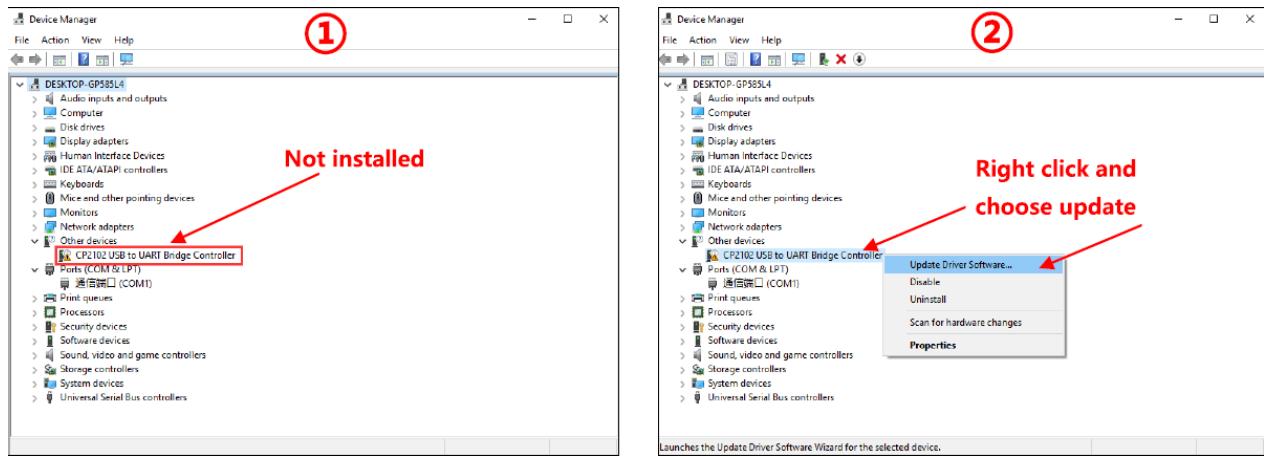
MEGA S 3D printer uses CP2102 chip for communication. So, it is necessary to install CP2102 driver so the printer could be recognized by PC.

Firstly, power on the machine and connect it to PC via USB cable.

CP2102 driver is copied in the SD card. "Files_English_MEGA S"-->"Driver _CP2102". There are two versions, Windows and Mac version.

For Windows, specifically, "CP2102xVCPIInstaller_x64" is for 64 bit system and "CP2102xVCPIInstaller_x86" is for 32 bit system.

Here we take Windows 7-64 bit PC system for example, while there is “Installation for Mac PC” in SD card for those who use Mac system. On PC, right click on “Computer”-->“Properties”-->“Device Manager”, and then follow the steps as shown in **Fig.18**.



Driver installation

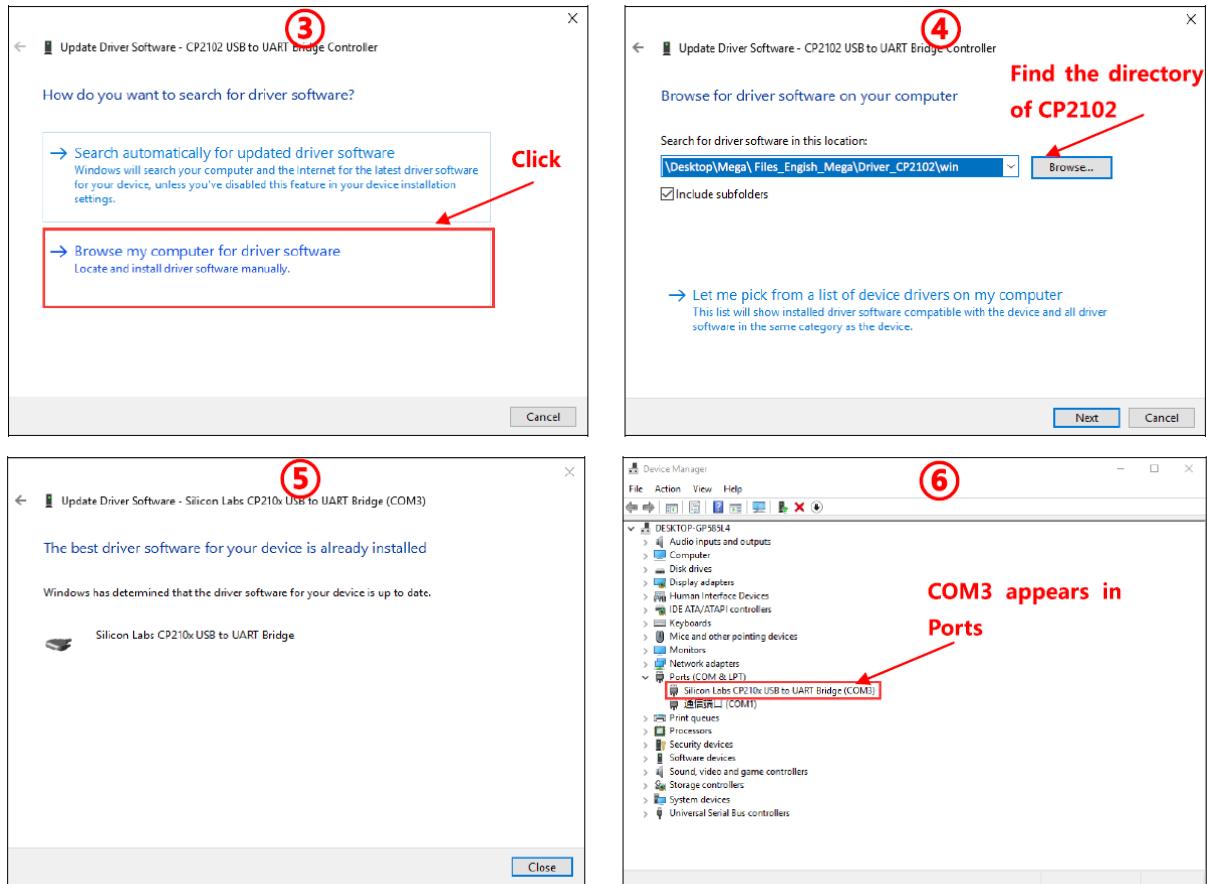


Figure 18

After successfully installation, a **COMx** would appear in the Ports of Device Manager, *x* is random (as here is COM3), customers will have their own COM*x* depends on their PC. This COM*x* port will be used for communication between the printer and PC later.

If the PC has been installed CP2102 driver before, then there should be a COM*x* in Ports of Device manager already.

Sometimes, even the driver is not installed properly, it may still show an abnormal COM*x* in Ports, please just uninstall the COM*x* and try to install the driver again.

Cura Installation

MEGA S 3D printer reads Gcode file and prints. It is necessary to convert 3D files (such as stl file) into Gcode files for machine to recognize. Software that convert 3D files into Gcode files is called slicing software.

Cura_15.04.6 is used for example here. Location of Cura: SD card--->"Files_Engish_MEGA S"--->"Cura"--->"Windows". Double click "Cura_15.04.6", and follow the procedures as shown in Fig.19:

Introduction to slicing software

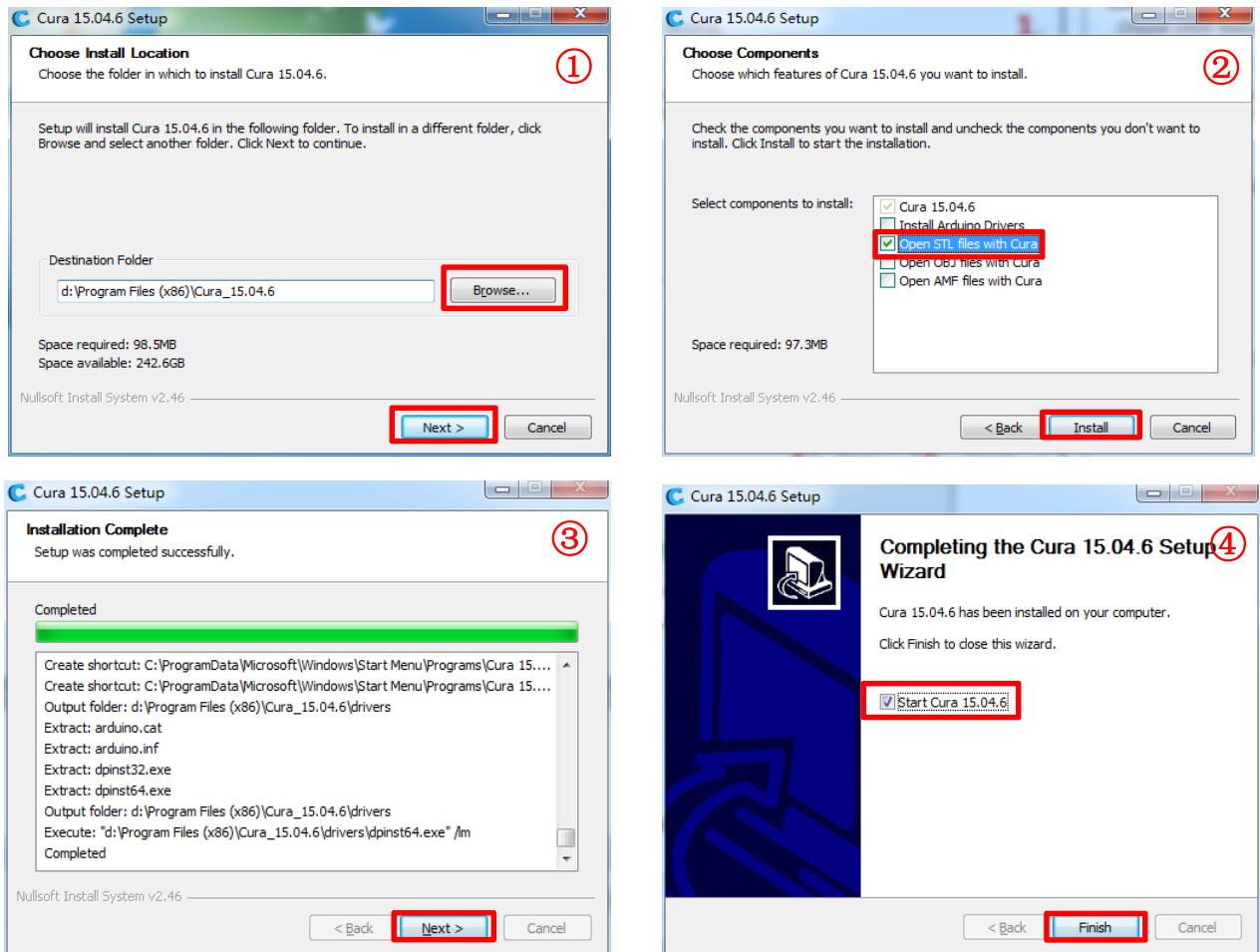
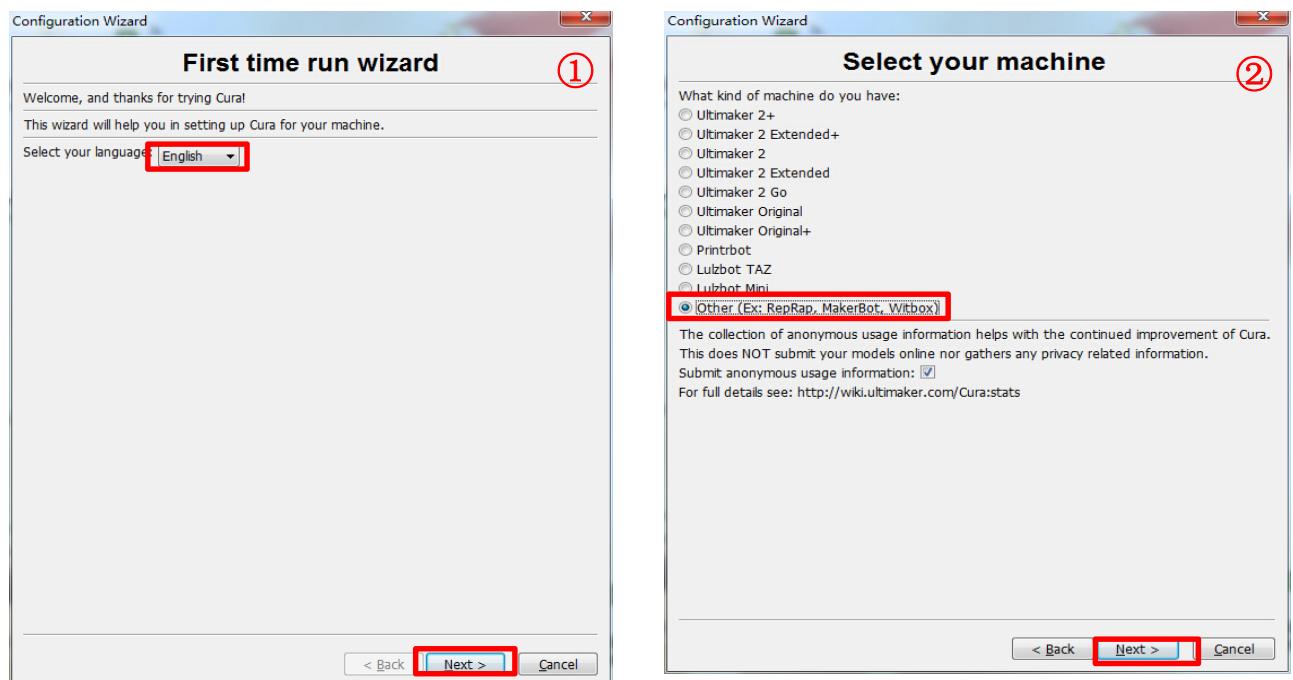


Figure 19

Next, before start Cura for the first time, there will be more settings about the language and machine types, as show in Fig.20.



Introduction to slicing software

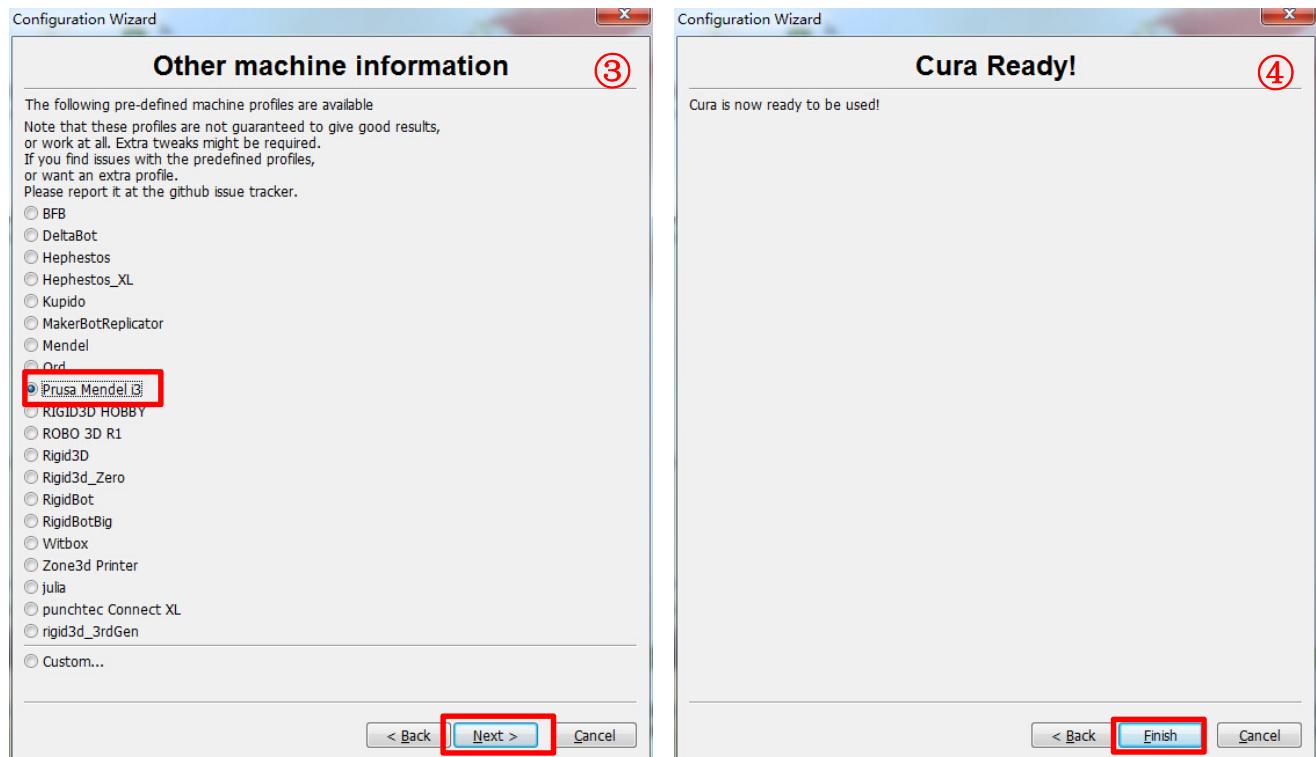


Figure 20

Upon finish, open Cura for the first time, there might be a default robot model appears, customers may click “File” ---> “Clear platform” to delete it.

Load 3D model into Cura

- (1) Clear the platform by clicking “File” ---> “Clear platform”
- (2) Load 3D files onto the platform by “File” ---> “Load model file...” . Files with extension such as “STL”, “OBJ”, “DAE” and “AMF” can be loaded. Model in grey means it is outside of the printable area and needs to be moved or scaled to fit in.

Manipulate 3D model in Cura

- (1) Zoom in/out: scroll the mouse wheel
- (2) Change viewing angle: right click the model, hold on and move the mouse
- (3) Position change: left click on the model, hold on and drag the model to move.
- (4) Rotate: single left click on the model and several icons will appear at the bottom left of the window (Fig. 21). Click the rotate button, 3 circles will surround the model. Rotate the model by moving the circle lines.

Introduction to slicing software

> Lay flat: it is very important to ensure the flat portion of the model is well attached to the platform. So, please use Lay Flat option everytime after rotating the model, as it will minimize the adhesion issues during printing. (Fig. 21)

> Reset: click it to return the model to the original orientation.

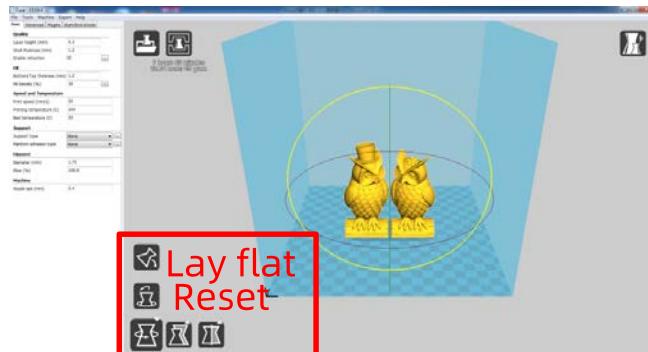


Figure 21

(5) Scale: to uniformly scale the model along X/Y/Z dimensions. To disable uniform scaling, click the lock in the lower section of the scaling window.

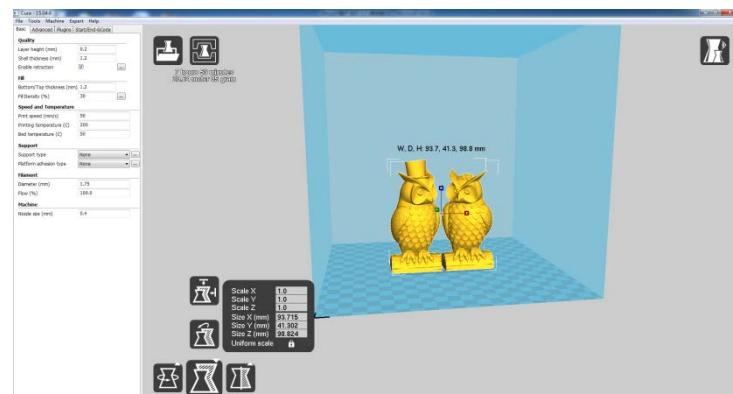


Figure 22

(6) View mode (Fig. 23): to view the model in different ways and helps to spotting issues before print starts. Such as “Layers” mode: to view the toolpath of the print head to check if there are skipped layers or gaps.

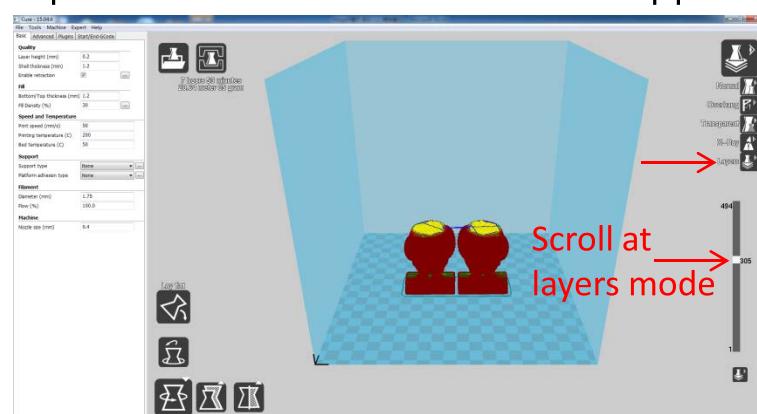


Figure 23

Cura settings

(1) Machine settings

Refer to Fig. 24, click “Machine”--->“Machine settings” to input those suggested parameters into the corresponding column. Please choose the Serial Port (COM) as shown in your PC→Device Manager→Port (refer to Fig. 18 -⑤, customers may have a different COMx other than the example COM3), and set the Baudrate to 250000. Those two parameters are essential for Cura to connect to the printer.

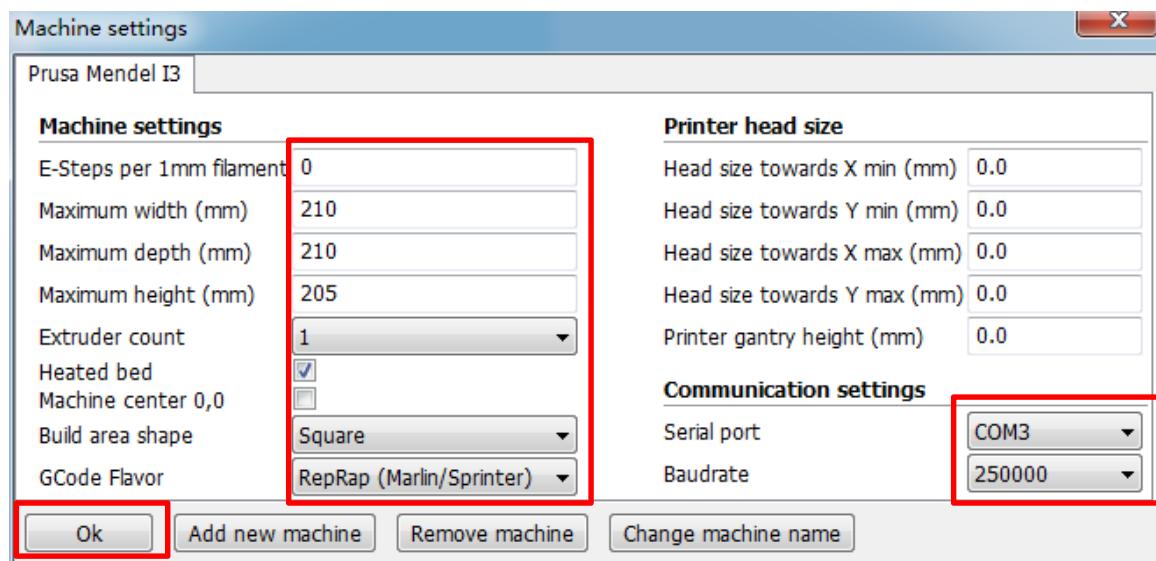


Figure 24

(2) Basic and Advanced options

Suggested “Basic” and “Advanced” setting are shown in Fig.25. Stay the mouse upon each box and there will be explanation for it. Those parameters are suggested for MEGA S 3D printer to print ANYCUBIC PLA filament. Generally, those settings are also compatible with other brand of PLA, but customers may need to fine tune the parameters to get the best results, for example, customers could try different ‘printing temperature’ based on the suggestion from a particular filament supplier. Especially, in order to get a good adhesion for the first layer, the ‘Bottom layer speed’ should not be too fast (20mm/s suggested).

Introduction to slicing software

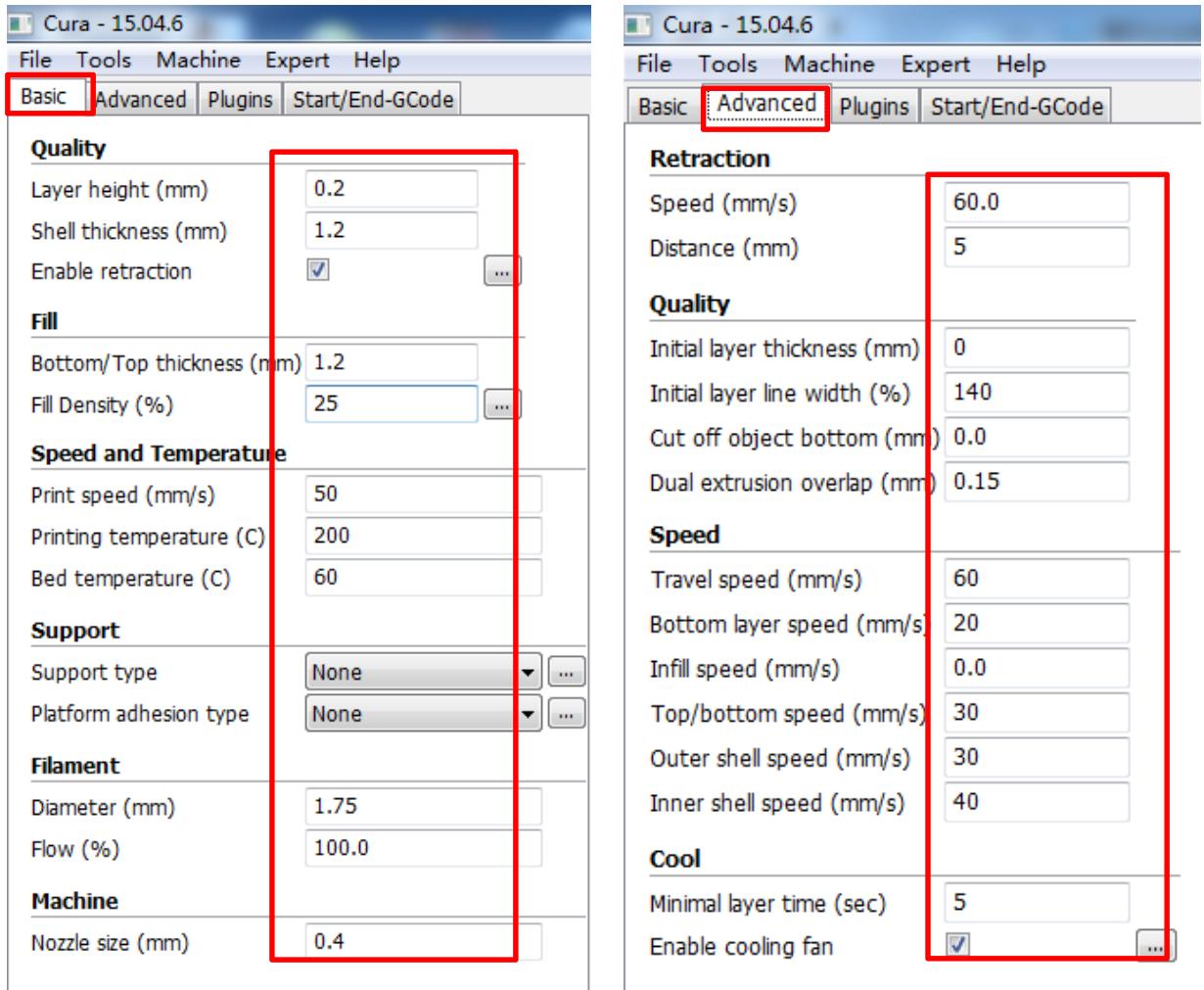


Figure 25

MEGA S is compatible with flexible filament, and we provide the settings as shown below if using ANYCUBIC flexible filaments (users may have to fine-tune the settings based on the actual printing conditions, and type of filaments, etc.). See next page.

Introduction to slicing software

Left Window (Expert Settings):

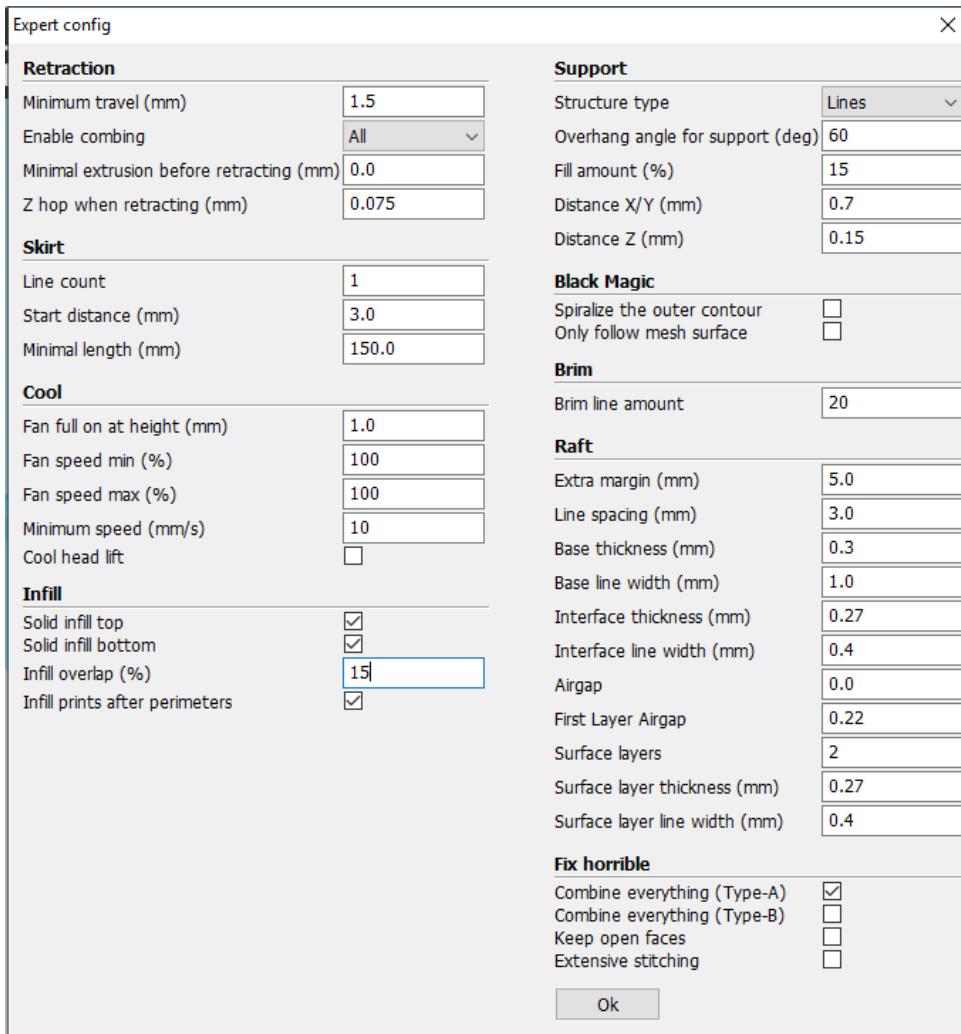
- Quality:**
 - Layer height (mm): 0.2, 0.8
 - Shell thickness (mm): 0.8
 - Enable retraction:
- Fill:**
 - Bottom/Top thickness (mm): 0.8
 - Fill Density (%): 10
- Speed and Temperature:**
 - Print speed (mm/s): 55
 - Printing temperature (C): 190
 - Bed temperature (C): 60
- Support:**
 - Support type: None
 - Platform adhesion type: Brim
- Filament:**
 - Diameter (mm): 1.75
 - Flow (%): 100.0
- Machine:**
 - Nozzle size (mm): 0.4

Right Window (Expert Settings):

- Retraction:**
 - Speed (mm/s): 91
 - Distance (mm): 6.4
- Quality:**
 - Initial layer thickness (mm): 0.2
 - Initial layer line width (%): 100
 - Cut off object bottom (mm): 0.0
 - Dual extrusion overlap (mm): 0.15
- Speed:**
 - Travel speed (mm/s): 80
 - Bottom layer speed (mm/s): 20
 - Infill speed (mm/s): 50
 - Top/bottom speed (mm/s): 30
 - Outer shell speed (mm/s): 15
 - Inner shell speed (mm/s): 30
- Cool:**
 - Minimal layer time (sec): 5
 - Enable cooling fan:
- Machine:**
 - Nozzle size (mm): 0.4

In the menu bar, select “Expert” → “Open expert settings”, and then set the parameters separately, as shown below:

Introduction to slicing software



(3) Plugins

*It is recommended for new user to leave the plugins as default (i.e. no plugins enabled).

Plugins are custom settings which will active at specific point during printing. There are two pre-loaded plugins with Cura: Pause at height and Tweak At Z. More plugins can be found via:

<http://wiki.ultimaker.com/Category:CuraPlugin>

As shown in Fig. 26, to enable one of the plugins, such as Pause at height, first click it and then click the drop-down arrow to enter the setting interface.

“Pause at height” will allow the printing to pause at a specified height, as well as where the print head would move to and how much filament to retract to prevent extruded filament blobs. So, customers could do filament change during printing.

Introduction to slicing software

“Tweak at Z” would allow custom changes at specified Z height.

Customers may decide the Z height or layer counts at which to make a change. Then there are more settings for how you would like to change, such as temperature, fan speed and print speed. Fine tune those for specific model would produce better results.

If wish to delete the plugins, stay mouse at the edge, hold the left button and drag mouse to show the Delete icon.

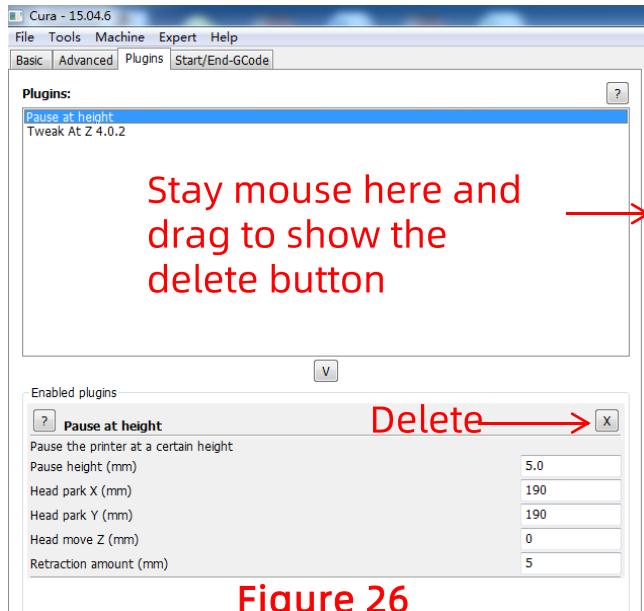


Figure 26

(4) Start/End-GCode

As shown in Fig. 27, custom Gcode allows for complex automatic printer movements and operations. By adding custom Gcode into the start or end of the Gcode file, customer could change how it prints. A detailed list of Gcode commands can be found via:

<http://reprap.org/wiki/G-code>

There will be explanation later about how to add command to start-gcode to achieve the function of resume from outage.

The screenshot shows the Cura software interface with the Start/End-GCode tab selected. A red annotation with the text "Delete" and a red arrow points to the "Delete" button located to the right of the "start.gcode" and "end.gcode" entries in the list.

```

;Sliced at: {day} {date} {time}
;Basic settings: Layer height: {layer_height}
;Print time: {print_time}
;Filament used: {filament_amount}m {filament}
;Filament cost: {filament_cost}
;M190 S{print_bed_temperature} ;Uncomment to
;M109 S{print_temperature} ;Uncomment to add
G21      ;metric values
G90      ;absolute positioning
M82      ;set extruder to absolute mode
M107     ;start with the fan off
G28 X0 Y0 ;move X/Y to min endstops
G28 Z0    ;move Z to min endstops
G1 Z15.0 F{travel_speed} ;move the platform
G92 E0          ;zero the extruded 1
G1 F200 E3       ;extrude 3mm of feed
G92 E0          ;zero the extruded 1
G1 F{travel_speed}
;Put printing message on LCD screen
M117 Printing...

```

Figure 27

Introduction to slicing software

Print online by Cura

After parameter settings, customer can print online by Cura with USB connection.

Click “File”--->“Print...” enter the printing popup window (Fig. 28). (If a simplified version of printing window appears, please click “File” --->“Preferences...” to choose the ‘Pronterface UI’ in the “Printing window type” drop-down menu)

Click “Print” icon when it is available after auto connect with the printer. Then the temperature would rise and it will start to print when reaching to the target temperature. Use tweezers to carefully get rid of the pre-extruded filament.

Note: If it fails to connect the printer in the popup “Printing window” (in a case that the “Print” icon is gray and unavailable), check with the COMx or Baudrate in “Machine settings”, and re-open the printing window to proceed.

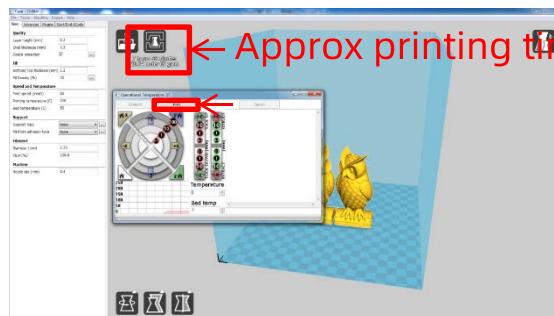


Figure 28

Save GCode in Cura

In Cura, click “File”--->“Save GCode...” to save the file to a desired directory. And it is highly recommended to save the Gcode in a SD card for printing offline.

>Re-open the Gcode file again in Cura to confirm that all slices of the model have been included by check it in the “Layers view” (refer to previous Fig. 23).

>The file name should only contain English letters, underscore and space. File name contains special characters could not be recognized by the printer. In order to let the printer better recognize the Gcode file in the SD card, you need to back up all the files in the SD card to the computer, and keep the SD card only for the Gcode file, please save all the Gcode files in root directory of the SD card.

Here shows the steps of printing offline (via SD card), print online please refer to **Page 22**.

1. As shown below, click “Tools”-->“Preheat” --> “Preheat PLA (for example)” **Fig.29**.



Figure 29

2. After the pre-heat is finished, please click on Home screen “Tools”-->“Filament” --> “Filament in”(**Fig.30**). The extruder motor will start to feed the filament into the hotend. There might be some excessive filament melt through the nozzle at high temperature, use tweezers to carefully remove it from the nozzle tip before print.



Figure 30

3. Insert the SD card into the SD card slot at the base. On Home Menu, Click “Print” to enter the files list. Click a exist file (e.g. “owl_pair”), and click “Print”(**Fig.31**). The machine will be sequentially heating the heated bed and nozzle and then print.



Figure 31

Printing

4. Upon finishing, the print head and heated bed will be automatically cooling down. Only remove the printed object from the heated bed when it is cooled completely. Pull the print platform to the front side, and use scraper to carefully remove the object as shown in Fig.32.

Never direct scarper to your hands.

Please be mindful that the nozzle and heated bed are still hot after operation.



Figure 32

5. MEGA S 3D printer equipped with **ANYCUBIC** (a novel print platform) which could be used for very long time without adding any masking tape, "hair spray" or glue stick. Customers only have to clean it by alcohol or similar after every few prints.

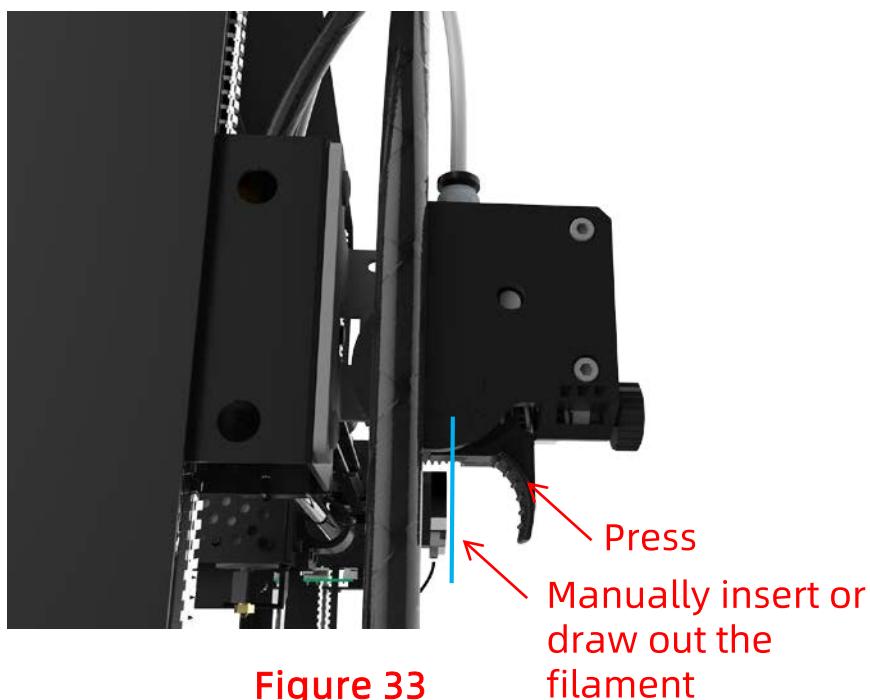
Suggested nozzle (print) temperature for PLA: 190-210 °C, ABS: 230-240 °C, Bed temperature for PLA: 60 °C, ABS: 80-100 °C. (it is suggested to disable the model cooling fan for ABS in Advanced settings of Cura)

After operation, do not immediately switch off the printer. Only turn it off after the nozzle cools to room temperature, because the heat sink still needs fan for cooling to minimize the risk of nozzle clogging.

1.Feed the filament: click via the Home menu: “Tools”--> “Preheat” --> “Preheat PLA (for example)”. After it reaches to the target temperature, press down the handle at the extruder as shown in **Fig.33**, and manually push the filament through the Teflon tubing till the hotend and there should be filament melt through the nozzle. Make sure the filament passes through the filament sensor first before reaching into the extruder.

For easier feed in the filament, it is suggested to cut off the bent tip before insert.

2.Remove the filament: at the Home menu, click “Tools”--> “Preheat” --> “Preheat PLA (for example)”. After it reaches to the target temperature, press down the handle at the extruder as shown in **Fig.33**, manually push in the filament firstly until seeing the filament melt through the nozzle, then quickly draw out the filament. The purpose of pushing in the filament is to minimize the risk of nozzle clogging.



Resume from outage

MEGA S allows resume print after accidentally power loss (This function only valid when print offline, via memory card only).

- As shown in Fig.34, Fig.35, in slicing software (i.e. Cura), it is required to place the model at the rear of the platform. Because during "RESUME", machine will home first and could touch/interfere with the unfinished object if the model was placed in the front area.
- For the first time of using this function, customers are required to add "G5" to the start.gcode, as shown in Fig. 36. Then, save the model as GCode file to the memory card by "File" → "Save GCode".

Note: ① "Resume from outage" is valid only for offline printing;

② Just type the "G5" when you use it for the first time, G5 will be automatically added later, without having to manually type it again.

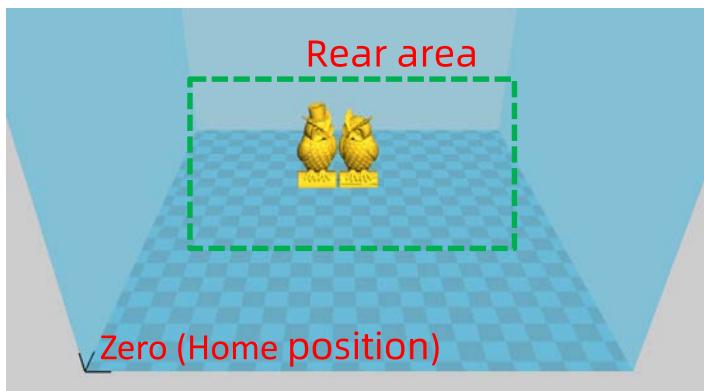


Figure 34

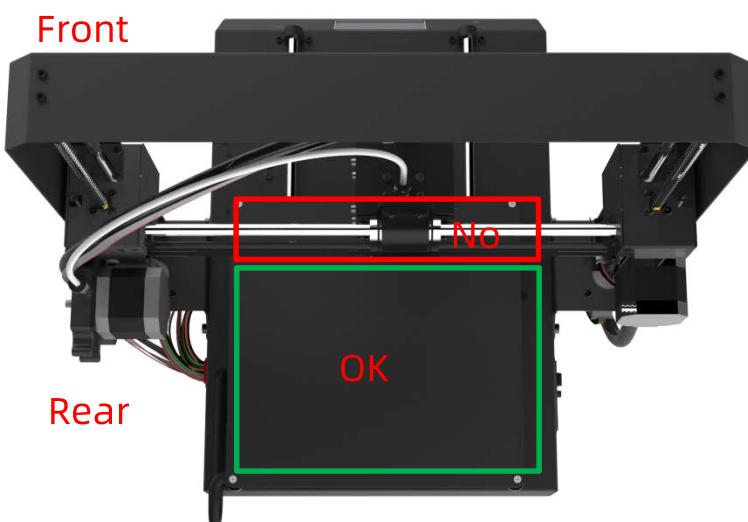


Figure 35

```

;Sliced at: {day} {date} {time}
;Basic settings: Layer height: {layer_height}
;Print time: {print_time}
;Filament used: {filament_amount}m {filament}
;Filament cost: {filament_cost}
;M190 S{print_bed_temperature} ;Uncomment to add
;M109 S{print_temperature} ;Uncomment to add
G21 ;metric values
G90 ;absolute positioning
M82 ;set extruder to absolute mode
M107 ;start with the fan off
G28 X0 Y0 ;move X/Y to min endstops
G28 Z0 ;move Z to min endstops
G1 Z15.0 F{travel_speed} ;move the platform
G92 E0 ;zero the extruded 1
G1 F200 E3 ;extrude 3mm of feed
G92 E0 ;zero the extruded 1
G1 F{travel_speed}
;Put printing message on LCD screen
M117 Printing...
G5

```

Figure 36

Resume from outage

3. During printing, if there is an accident power loss, the print will stop immediately. But after power comes back, customers could choose “Print” → select the unfinished file → “RESUME”(Fig.37), machine will home first and continuing on the unfinished object.



Figure 37

Note:

- ① In order to get smooth surface, use tweezers to carefully remove the excessive filament at nozzle before continuing print upon the last point.
- ② Do not move Z axis after power off otherwise resume will be invalid.
- ③ MEGA S supports resume from outage only when print offline
- ④ This function is developed based on Cura. We could not guarantee this function compatible with other slicing software.
- ⑤ Due to the differences of filaments, temperature, extrusion, etc...we could not guarantee a perfect surface at the point of “RESUME”, especially for small objects.

1. Motor shaking or abnormal sound

- ① The corresponding end stop could not be triggered when Home, check the wirings, and inspect any obstacles by manually moving the corresponding axis
- ② The motor cable are not connected properly, check each connection and then inspect the cable routing for any faults

2. File not printing or memory card failure

- ① Remove the memory card and insert into PC. Open the GCode files using text editor (eg. Notepad), and inspect if GCode is readable or not. If files contains of multiple “ÿÿÿ” symbol, then file has been corrupted. Try reformatting the memory card to FAT32 format and reloading the GCode file
- ② Memory card is not readable, ensure file name does not contain special characters or Change memory card
- ③ Touch screen freeze, reboot the machine and try again

3. No extrusion or extrusion motor knocking

- ① Ensure that the nozzle temperature has been set to match the filament
- ② Filament tangled on spool
- ③ Not enough cooling for the hotend
- ④ Nozzle clogged please try to replace it or clean it
- ⑤ Teflon tubing has been tangled, squeezed or bent

4. Filament leaking

Nozzle or throat tube is tightened properly, try to fix/change it after cooling

5. No sticking to the bed

- ① Print too fast at the bottom layer speed, reduce it to ~20mm/s
- ② Ensure that the print platform is clean (use alcohol if necessary)
- ③ Check if the bed is proper leveled
- ④ Add a brim or raft to the model in slicing software
- ⑤ Check the bed temperature matches the filament

6. Warping/curling of the printed object

- ① Check the bed temperature matches the filament
- ② Check the infill % of the GCode. The higher the infill, the more likely to warp
- ③ Add a brim or raft to the model in slicing software.

7. Layer shifting

- ① Print head moving too fast, slow down the print speed.
- ② Check X/Y belt and the driving wheel and ensure they are properly installed.
- ③ Grease the rods and check all nuts and bolts remain tightened.

8. Freezing screen

- ① Inspect if the touch screen has been pressed by the metal frame at the edge
- ② Check if screen has cracks, if so, please contact us at
www.anycubic.com

9. T0 sensor abnormal

- ① Check the wiring of the hotend and ensure a good connection
- ② Check if there is any pins bent inside the connector

10. Print head move abnormal

- ① Check if choosing the right machine type in slicing software
- ② Check if any plugins in the slicing software

11. Print stopped halfway

- ① Check if the GCode file is corrupted
- ② Delete plugins in the GCode file
- ③ Use print offline mode (memory card) instead of print online via data cable

Thank you for purchasing ANYCUBIC products! Under normal usage and service, the products have a warranty period up to one year. Please visit ANYCUBIC official website(www.anycubic.com) to report any issues with ANYCUBIC products. Our professional after-sale service team would response within 24 hours and solve the issue.